Cervenka Consulting Ltd.
Na Hrebenkach 55
150 00 Prague
Czech Republic
Phone: +420 220 610 018
E-mail: cervenka@cervenka.cz
Web: http://www.cervenka.cz

ATENA Program Documentation
Part 4-5

Tutorial for thermal analysis

Coupling of thermal and stress analyses

Written by

Jan Červenka, Dobromil Pryl

Prague, Aug. 20, 2009

Copyright © 2000-2009 by Cervenka Consulting.

Trademarks:

Microsoft and Microsoft Word are registered trademarks of Microsoft Corporation.

TABLE OF CONTENTS

1. INTRODUCTION 1

2. THERMAL ANALYSIS 1

3. STRESS ANALYSIS 8

4. POSTPROCESSING 12

4.1 Postprocessing in GiD 13

4.2 Postprocessing in ATENA 3D 16

5. CONCLUSIONS 16

6. LITERATURE 18

1. INTRODUCTION

This document describes an example of rotationally symmetrical vessel subjected to thermal loading. The analysis is performed using the programs ATENA and GiD. ATENA is used for thermal and static analysis and the program GiD is used for data preparation and mesh generation.

The programs GiD and ATENA can be installed using the standard ATENA installation. At the end of the installation the user must select the installation of GiD and ATENA-GiD interface. After that your computer should be ready to run the example problem described in this document.

2. THERMAL ANALYSIS

First the program GiD is started. The recommended version is 9.0.2 or newer (the oldest supported version is 7.7.2b). After starting GiD, the user should open the example analysis "ATENA Science\ATENA-GiD\Tutorial.Temperature2D\PipeBTemp.gid". This is an existing model demonstrating the combination of thermal and stress analysis. This problem is using the problem type: AtenaV4 | Transport. It represents a section of a pipe wall with thickness of 0.23 m and internal diameter of 1 m. Taking advantage of the symmetry, only a quarter of the whole crossection is modeled. The geometry of the model is shown Figure 1 and the numerical model is shown in Figure 2. Details about ATENA-GiD interface and associated problem types for ATENA can be found in the manual [6]. The same mesh size is used for thermal and static analysis. This is however not a strong requirement. The thermal loading can be exchanged also between models with totally different meshes.

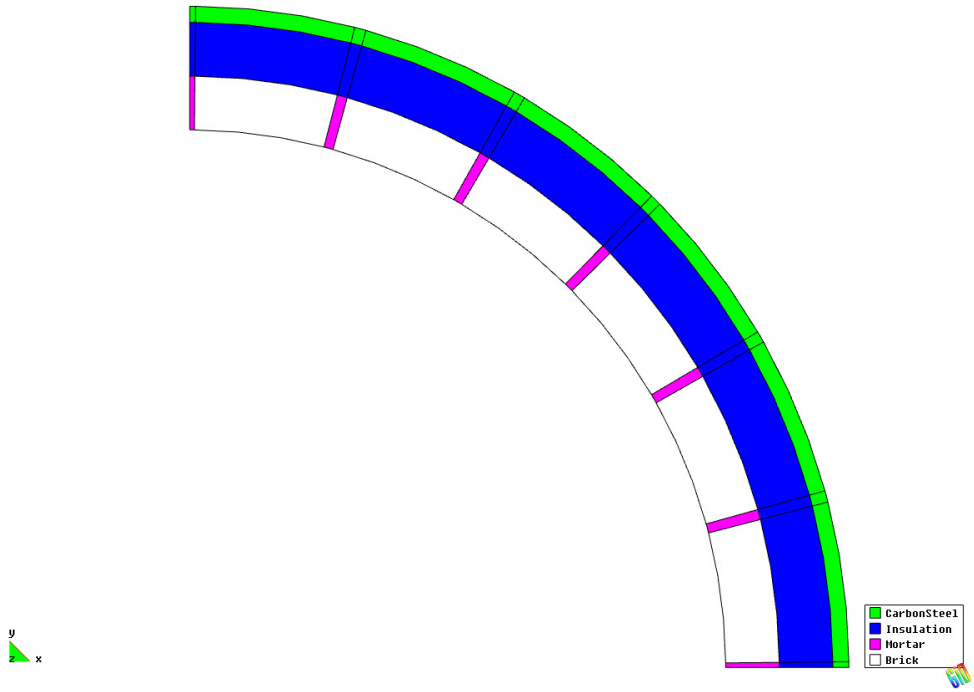

Figure 1: Geometry and material properties of the axisymmetrical pipe model.

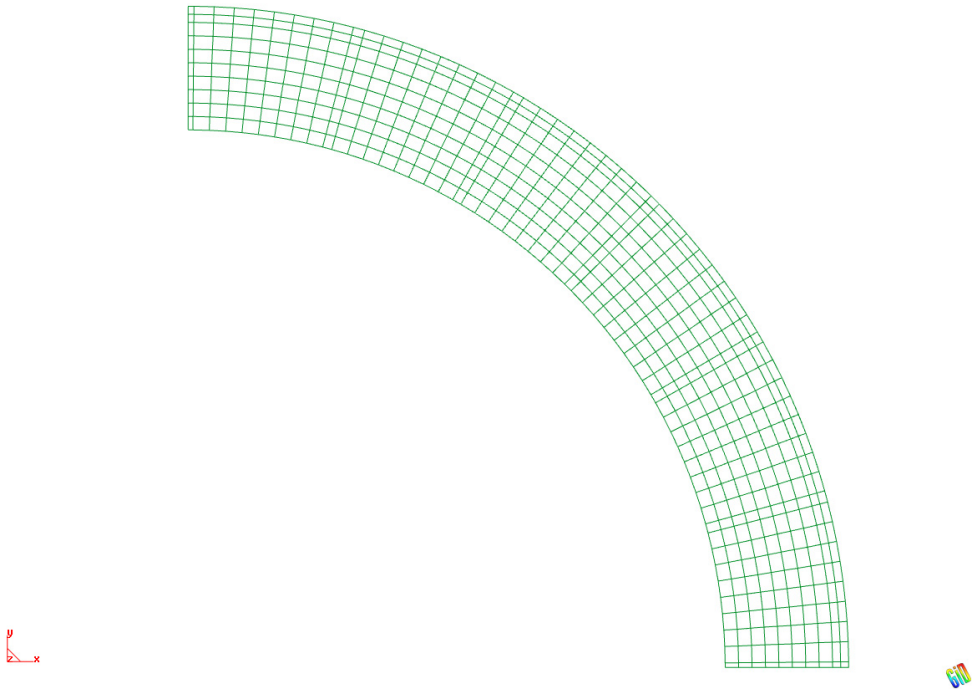

Figure 2: Numerical model, finite element mesh.

The loading is subdivided in 3 intervals. In the first interval 12 load steps are defined with boundary conditions as described in Figure 3. In each step the temperature on the outer surface is increased by 1 °C (see Figure 3). The temperature in the exterior is increased up to 37 °C, starting from the initial uniform 25 °C. Each step in this interval represents 3000 seconds, thus the whole interval covers the period 0-36000 seconds (0-10 hours).

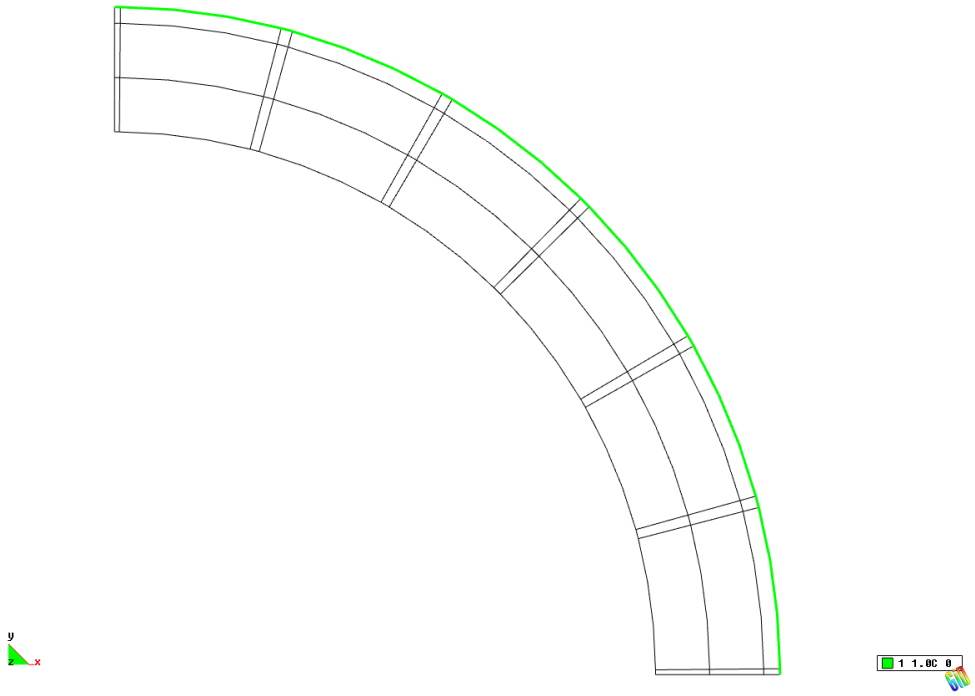

Figure 3: Boundary conditions for the interval 1.

In the subsequent interval 2, the temperature at the outer surface is kept constant. This interval contains 10 steps 2880 seconds long. This means the whole interval spans the time period from 36000-64800 seconds (10 hours-18 hours).

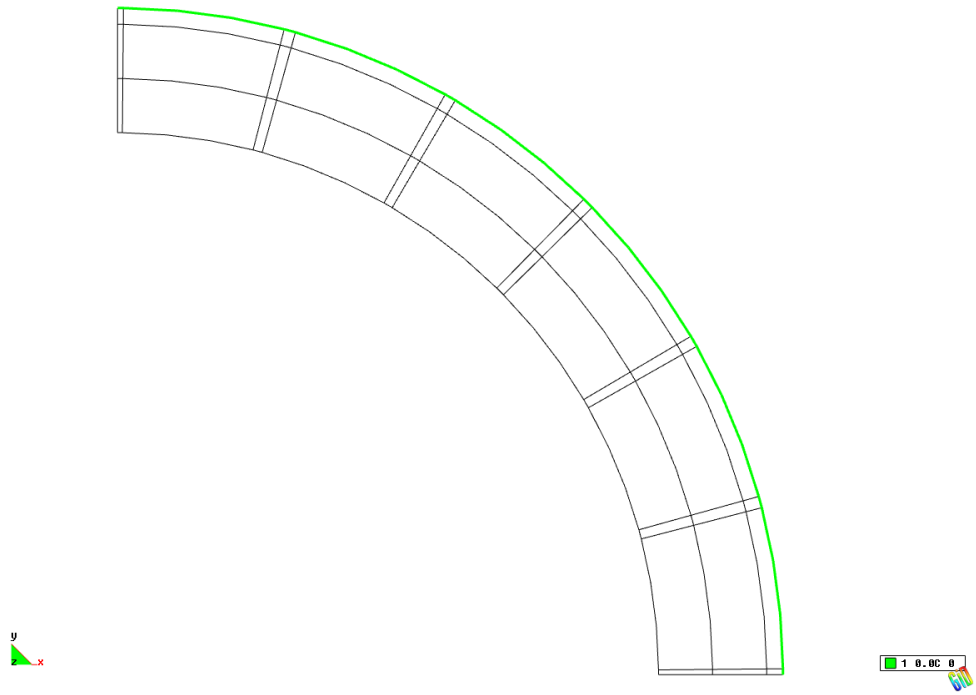

Figure 4: Boundary conditions for the interval 2.

In the last interval 3, the outer surface is cooled back to 25 degrees. This interval contains 12 steps 1800 seconds long. This means the whole interval spans the time period from 64800-86400 seconds (18 hours-24 hours).

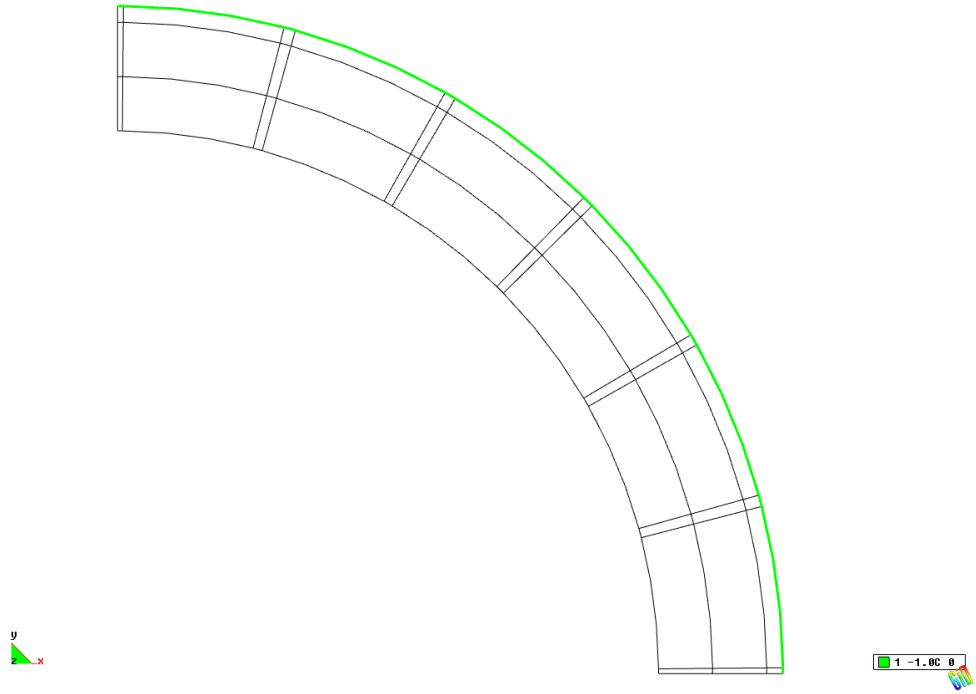

Figure 5: Boundary conditions for the interval 3.

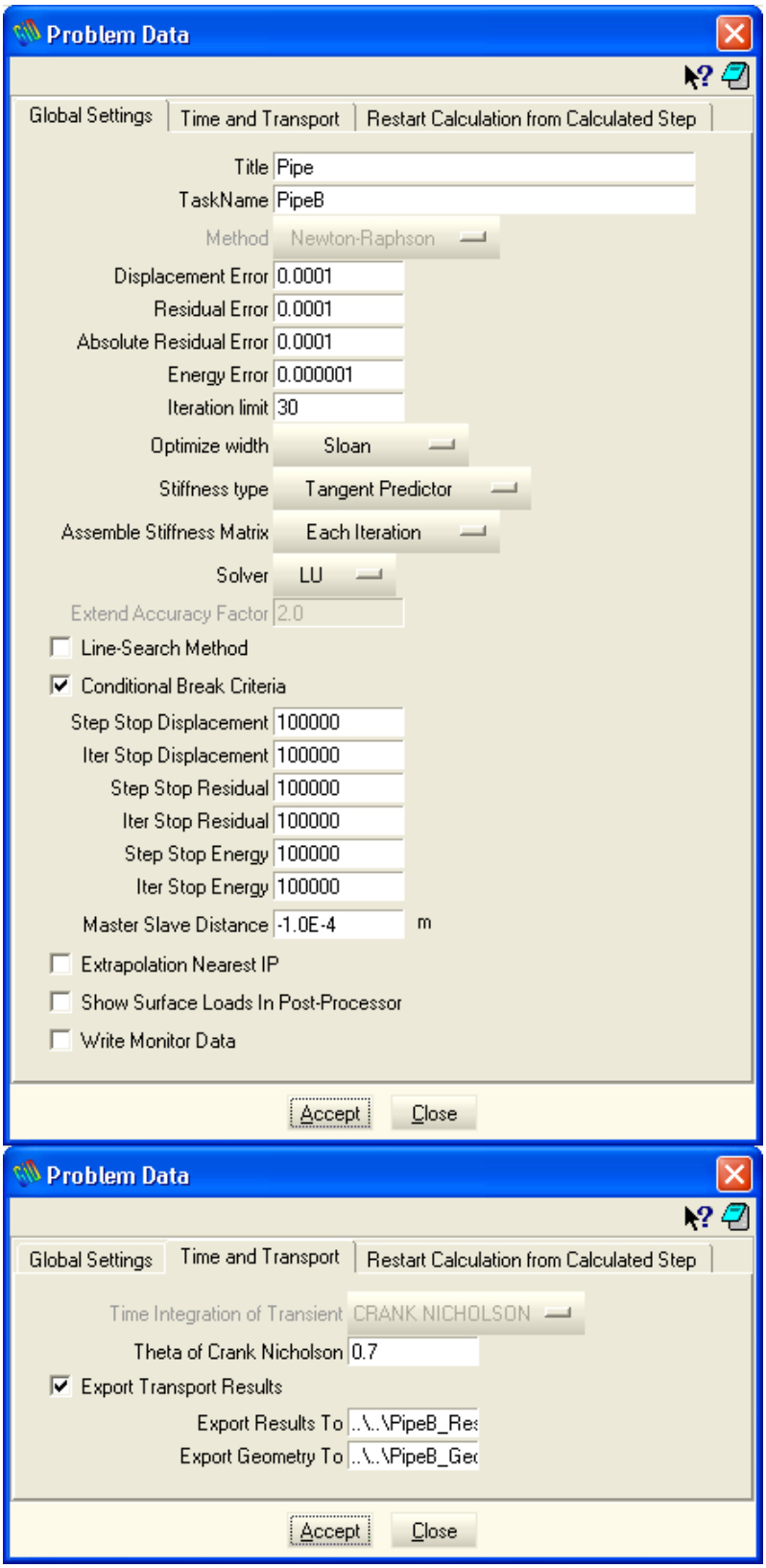

Figure 6: Problem data dialog including the definition of temperature exchange files with stress analysis.

The problem data dialog that is shown in Figure 6 can be opened via the menu item Data | Problem data. This dialog can be used to define the basic parameters for the thermal analysis. The most important fields can be found at the bottom of the Time and Transport tab, where the names of two files are to be specified. These files are used for exchanging

the temperature fields with the subsequent stress analysis. By default, the files would be stored in the AtenaTransportCalculation subdirectory of the main problem directory, inserting "../../" before the names writes them into the Tutorial.Temperature2D directory.

The ATENA calculation is started from the menu ATENA | ATENA analysis or by clicking the calculator icon . This will start AtenaWin program which is a graphical interactive environment for the execution control of ATENA finite element core module. More details about the usage of this program can be found in the corresponding ATENA manual [5].

After executing ATENA analysis the following window appears on the user's computer (see Figure 7).

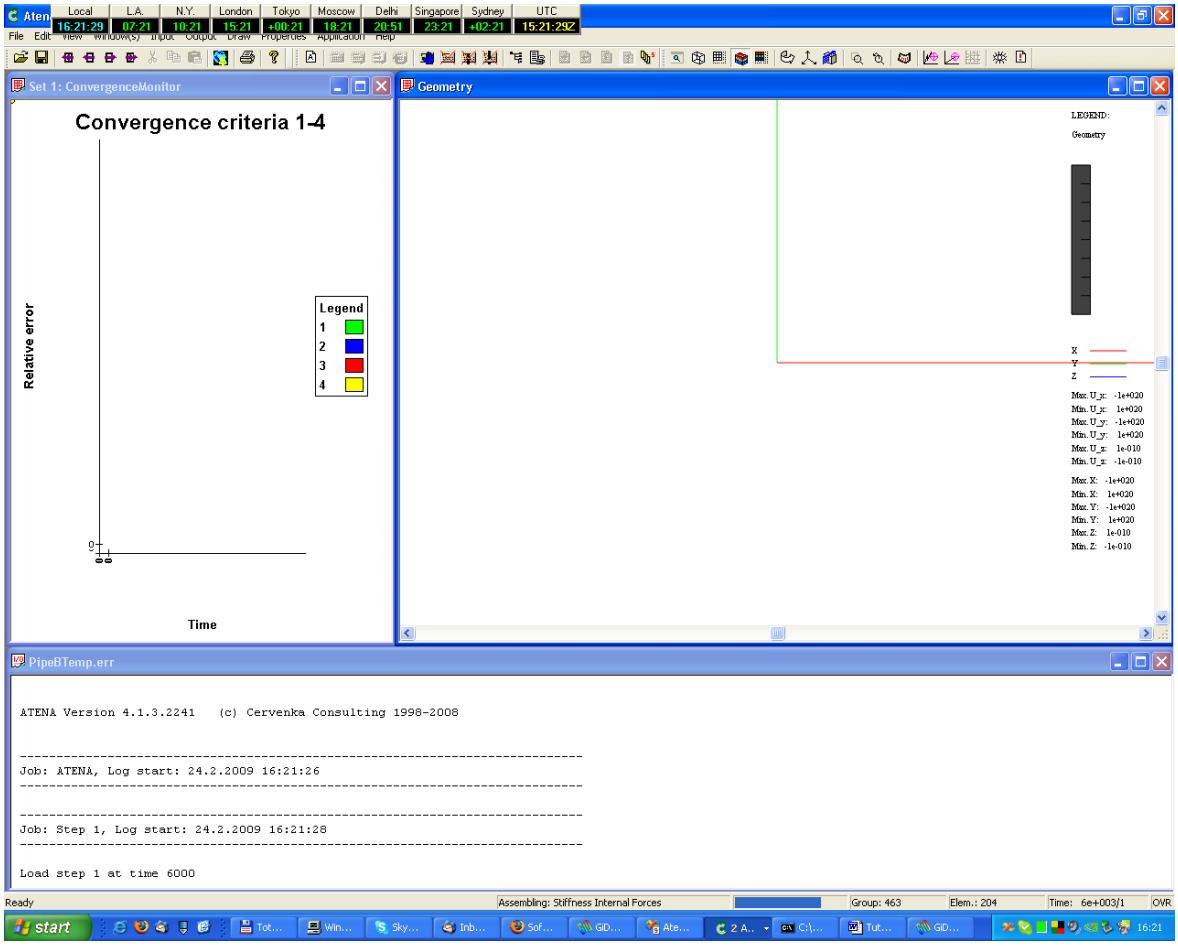

Figure 7: The main AtenaWin window after its activation from GiD.

The ATENA analysis is started automatically (or by clicking the button). This starts the thermal analysis in AtenaWin environment. In order to visualize the development of the temperature field the button can be selected and this will open a dialog that is shown in Figure 8. In this dialog various variables can be selected for display. The temperature fields can be displayed by selecting "CURRENT_PSI_VALUES" | Temper.

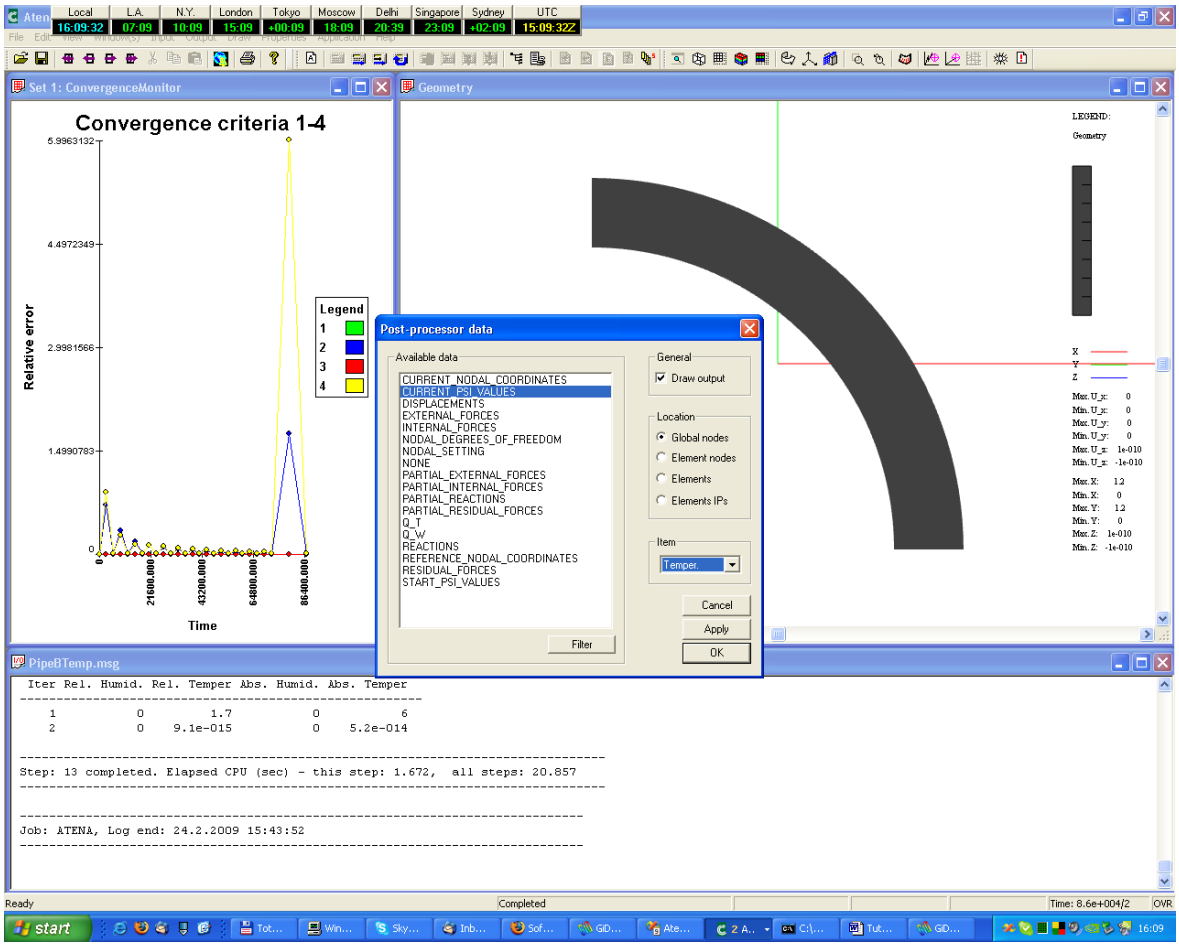

Figure 8: The selection of temperature display in AtenaWin.

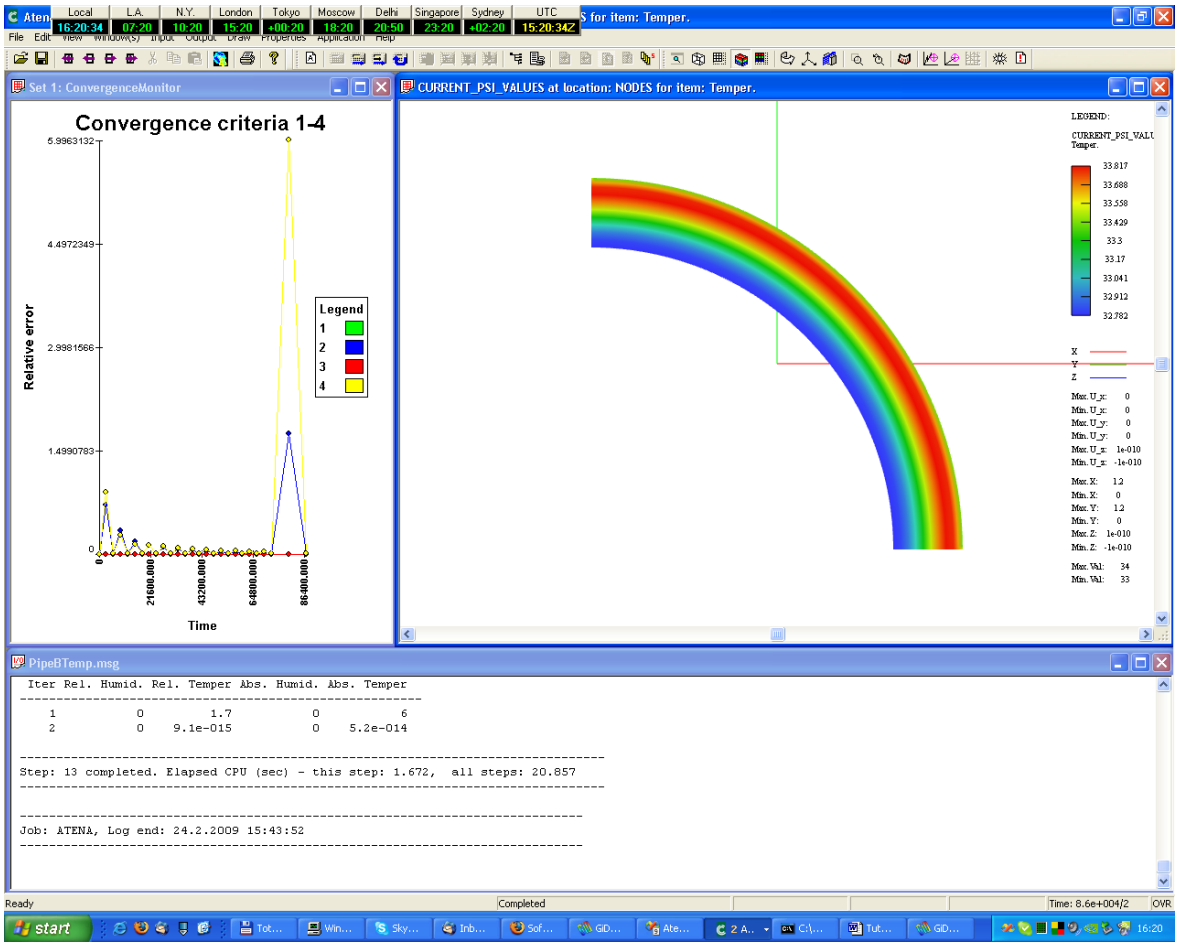

Figure 9: The runtime display of temperature field during the thermal analysis in ATENA.

After the thermal analysis is completed AtenaWin can be closed. All resulting files are stored in the subdirectory AtenaTransportCalculation of the PipeBTemp.gid directory. In this subdirectory the following files will exist after the completion of the thermal analysis:

PipeBTemp.inp	ATENA input file created by GiD and used by AtenaWin
PipeB.00xx	Binary result files created by ATENA during the thermal analysis.

These two files are created in the Tutorial.Temperature2D directory:

PipeB_Results.thw	Saved temperatures to be used by stress analysis.
PipeB_Geometry.bin	Saved geometry to be used for the interpolation of temperatures in the stress analysis.

3. STRESS ANALYSIS

After the thermal analysis is completed AtenaWin can be closed and the stress analysis can be performed using the calculated thermal fields. A new GiD problem must be created or the existing problem PipeBStatic.gid can be used. This model defines the input for stress analysis of the same pipe wall as was used in the thermal analysis.

The geometrical model and boundary conditions are shown in Figure 10.

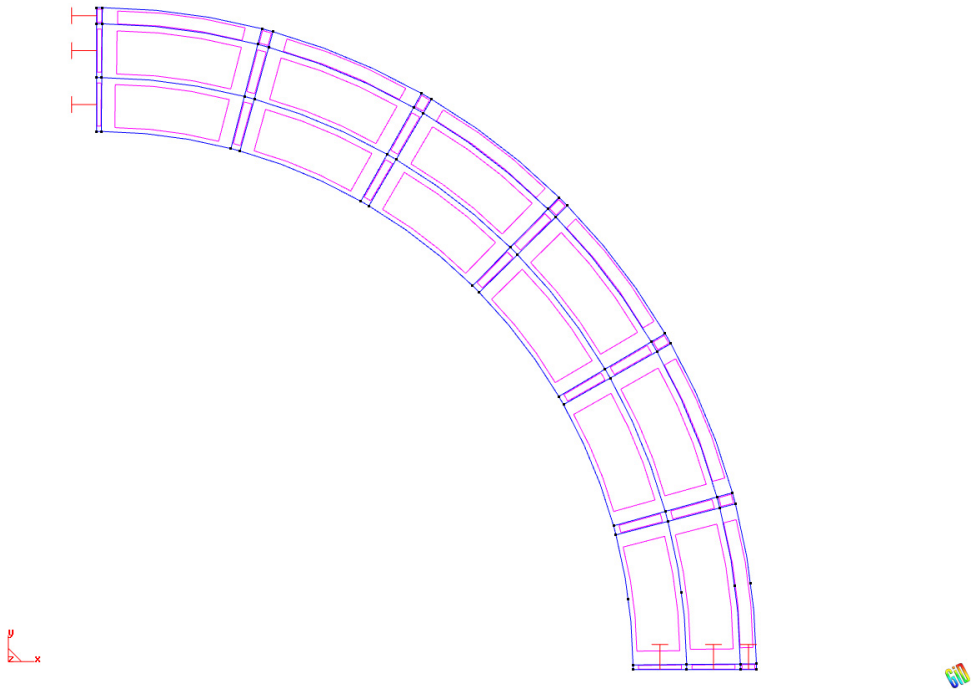

Figure 10: Geometrical model and boundary conditions for stress analysis.

In order to be able to utilize the thermal fields calculated during the thermal analysis the appropriate import files must be specified in the problem data dialog that is activated from the menu Data | Problem data and is shown in Figure 11. This information is located in the bottom 2 input fields where appropriate file names are specified including their path.

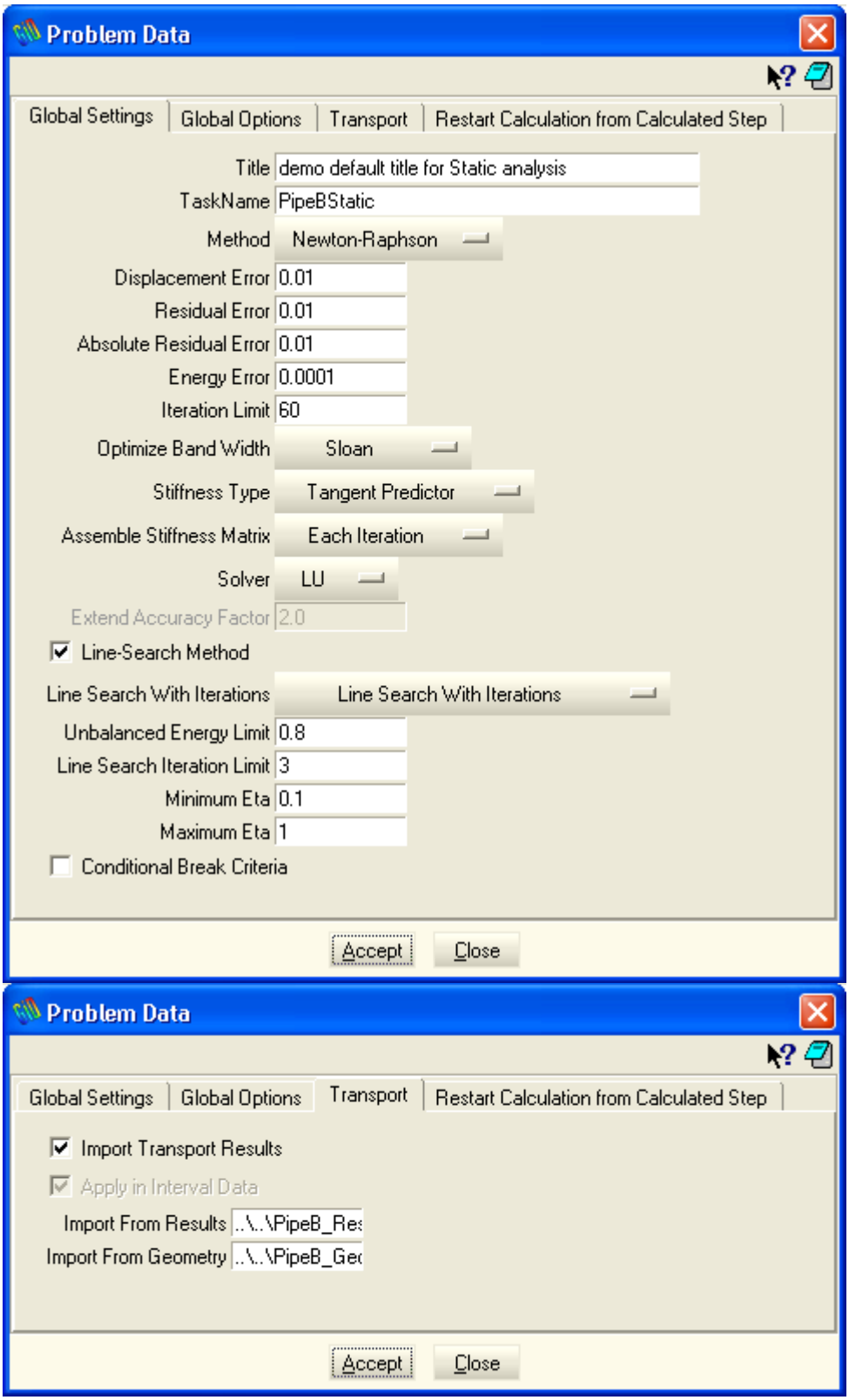

Figure 11: Problem data dialog including the definition of temperature exchange files.

The loading history is specified in a single interval. The interval is divided into 50 load steps. In each step the temperature difference of 720 seconds from the thermal analysis is applied (Figure 12). So the interval spans the period of 10 hours, i.e., it only covers the heating phase.

The Transport Import switch set to Interval Beginning means the thermal analysis results are only imported once and the temperature values are interpolated for each static analysis step. For complex temperature histories, it can be changed to Each Step.

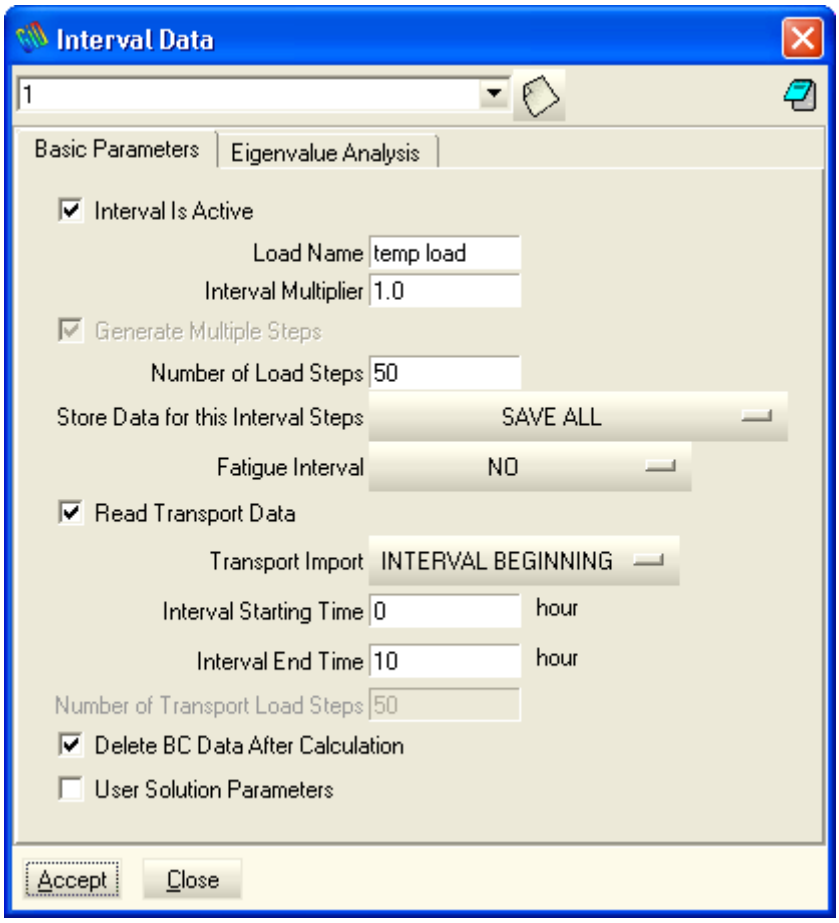

Figure 12: Interval data for the interval 1.

The stress analysis is again started by selecting the menu item ATENA | ATENA analysis or the icon. This starts the AtenaWin program. The analysis is started automatically or by clicking the button. This starts the stress analysis in AtenaWin environment. In order to visualize the development of the various data fields, the button can be selected and this will open a dialog that is shown in Figure 13. In this dialog, various variables can be selected for display. The contour areas of crack width can be displayed by selecting Elements | "CRACK_ATTRIBUTES" | COD1. The resulting computer screen in shown in Figure 14.

The imported temperature values can be displayed as ELEM_TOTAL_TEMPERATURE | TotalTemp. Please note that only the difference from the reference (initial) temperature is displayed in static analysis.

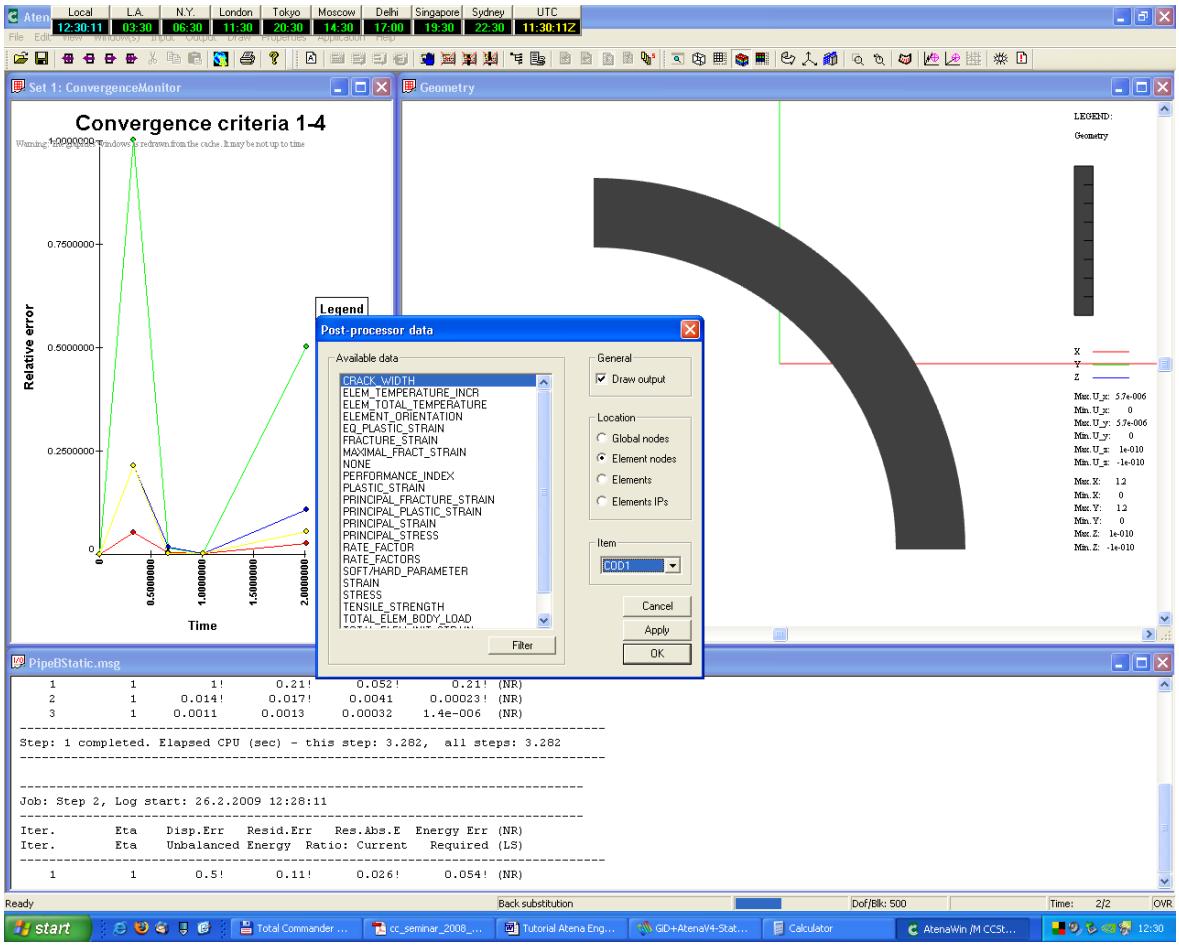

Figure 13: Execution of static analysis in AtenaWin and the selection of crack opening display.

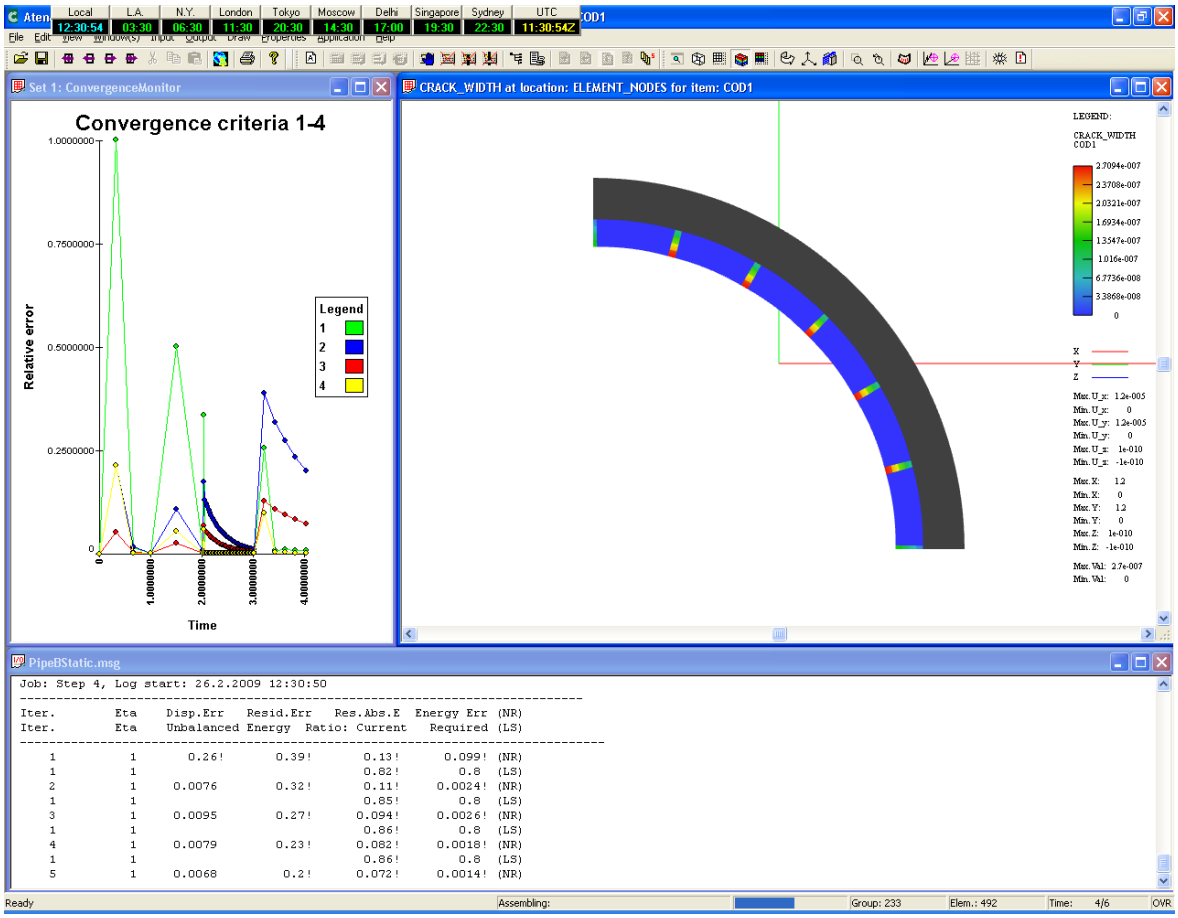

Figure 14: Execution process of stress analysis in AtenaWin showing the crack opening displacements.

After the completion of the analysis the AtenaCalculation subdirectory of PipeBStatic.gid contains the following files:

PipeBStatic.inp	ATENA input file created by GiD and used by AtenaWin
PipeBStatic.00xx	Binary result files created by ATENA during the stress analysis.

4. POSTPROCESSING

Postprocessing can be done either in AtenaWin, GiD, or Atena 3D. Normally, AtenaWin displays the current step that is analysed. After the analysis is finished, the last step remains in AtenaWin memory and can be visualized and further post-processed. In case the user wants to post-process results from other load step, the corresponding step results file is to be opened using the command Application | Restore FE Model From. The step data file name is "*task_name*.00xx", where *task name* is the name of the current task as given in the problem data dialog (see Figure 11) and *00xx* represents the load step number which is to be post-processed.

(Alternatively, the new text window is to be opened using the menu item File | New. Please note that a text window must be highlighted in order to another text window. If

graphical window is the active one, a new graphical window will be opened. Into the new text window the following command shall be written:

RESTORE FROM "*task_name*.00xx")

4.1 Postprocessing in GiD

To be able to postprocess the results in GiD, the result quantities must first be made available by selecting them in the Data | Problem Data | Post Data dialog, see Figure 15.

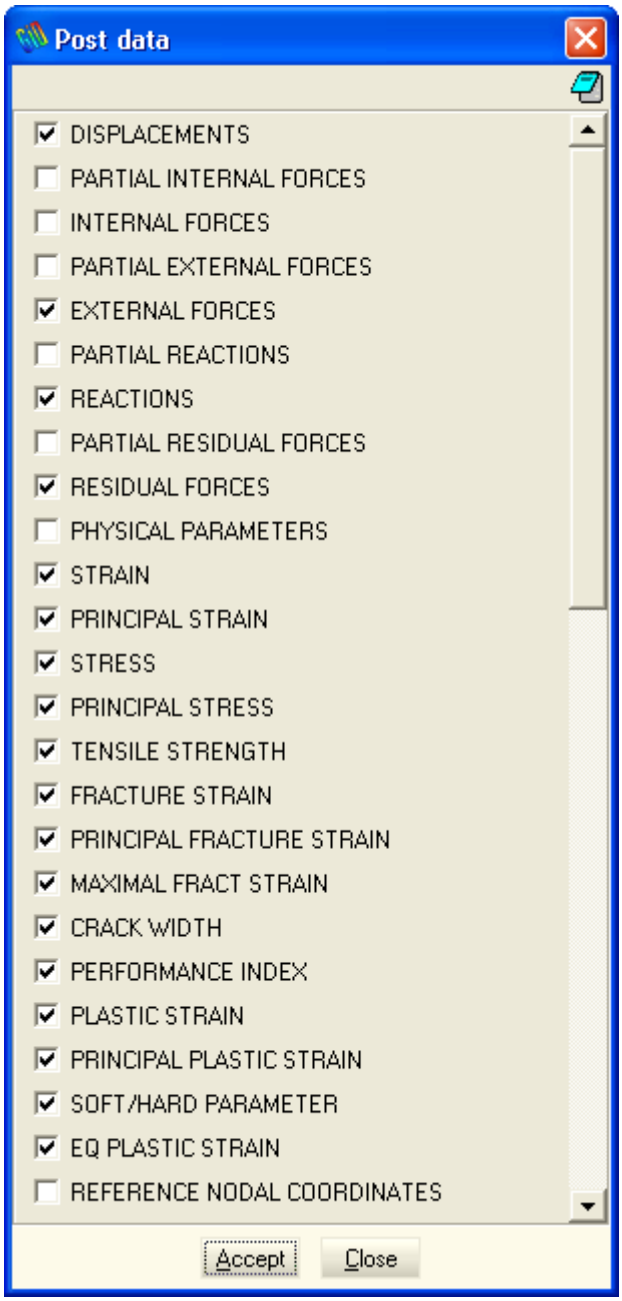

Figure 15: Selecting results for postprocessing in GiD.

The menu command ATENA | GiD Post-processing or the icon toggles GiD between pre- and postprocessing. A warning about non-existent .res file may appear, then a console windows is started and the results are converted into a format readable by GiD, see Figure

16. The conversion can take a few minutes depending on model size, number of load steps, and the number of quantities selected for postprocessing. The results are stored into the *AtenaResults.flavia.res* file in the *AtenaCalculation* respectively *AtenaTransportCalculation* directory for static analysis respectively transport analysis. This file can be opened in GiD by the File | Open command (see Figure 17).

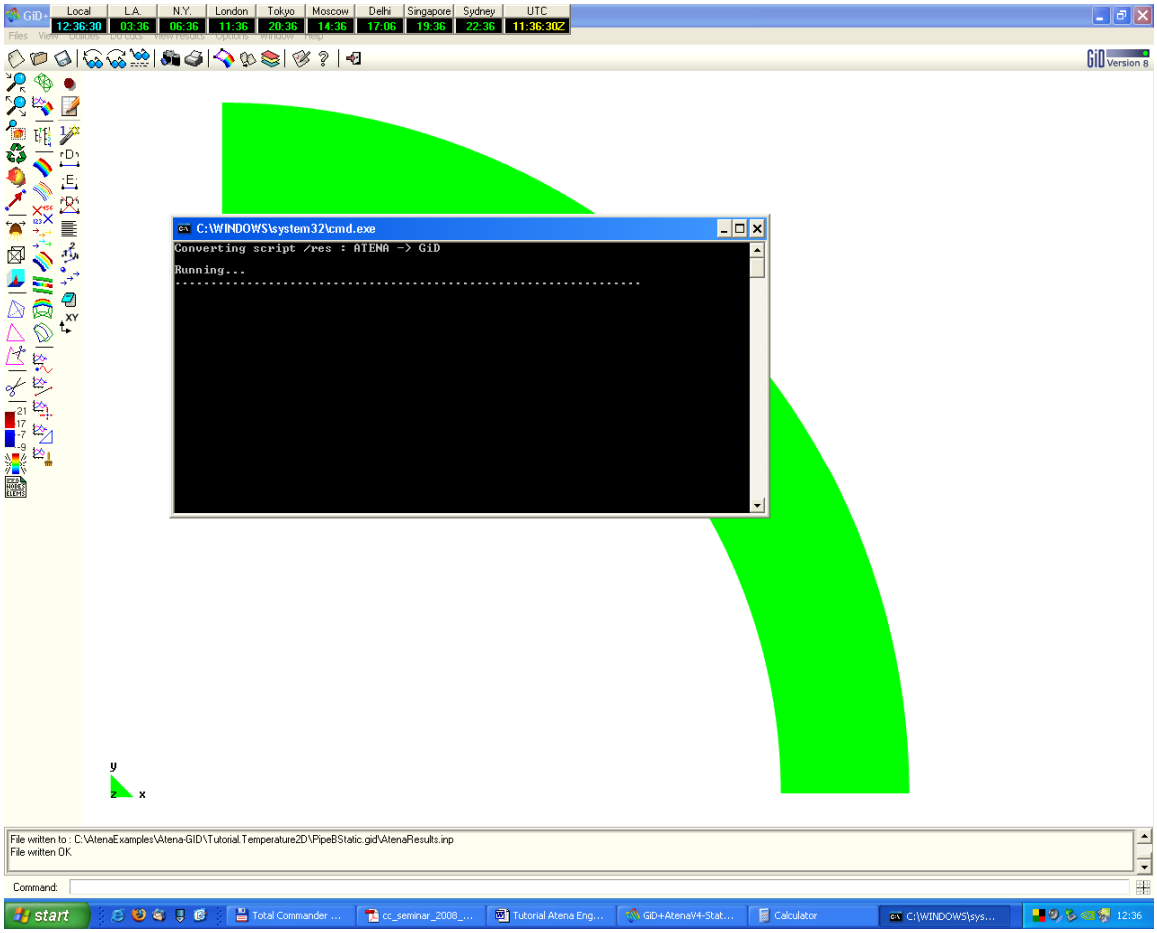

Figure 16: Importing results for postprocessing in GiD.

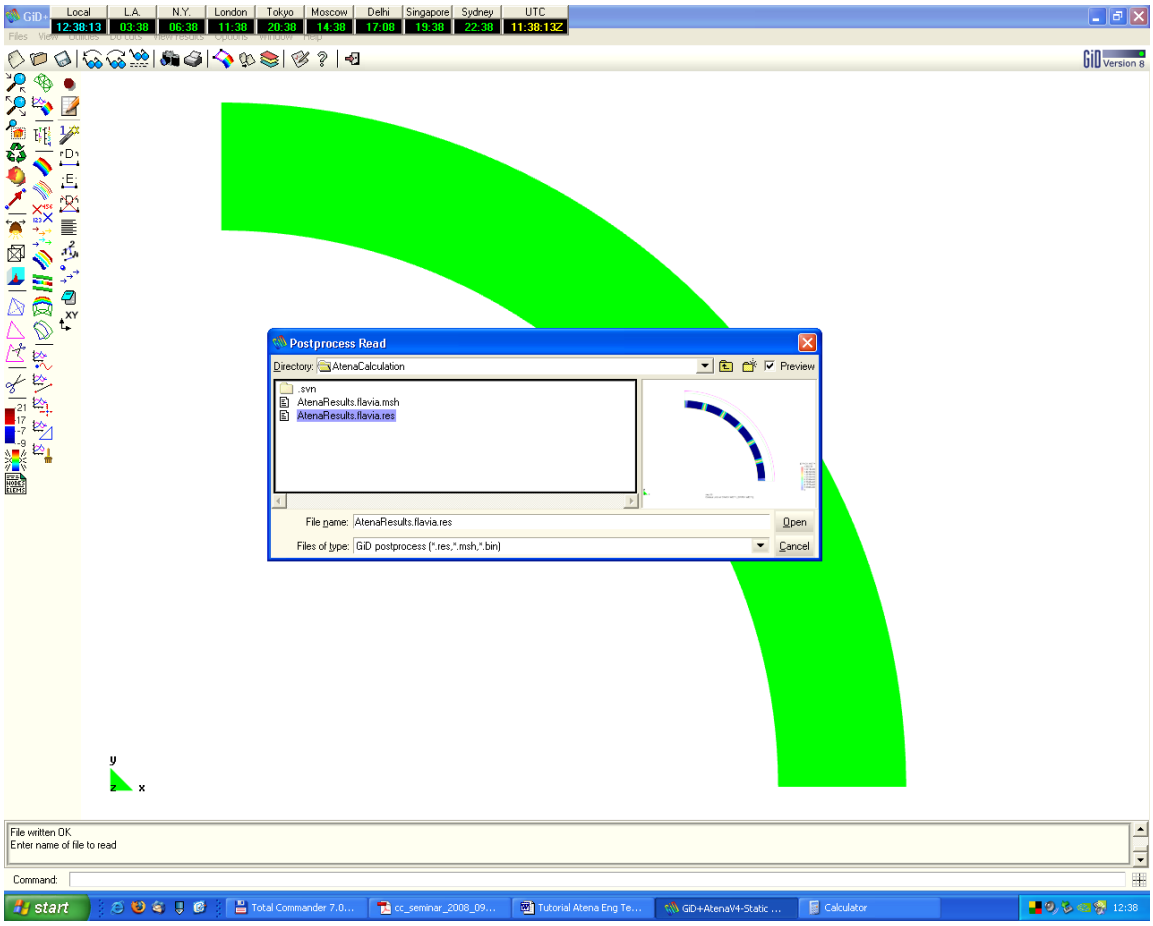

Figure 17: Opening results for postprocessing in GiD.

The results can be then postprocessed. Figure 18 shows crack width as contour lines, which can be selected by the menu command View results | Contour lines | CRACK WIDTH | COD1. The command can also be accessed from the icon.

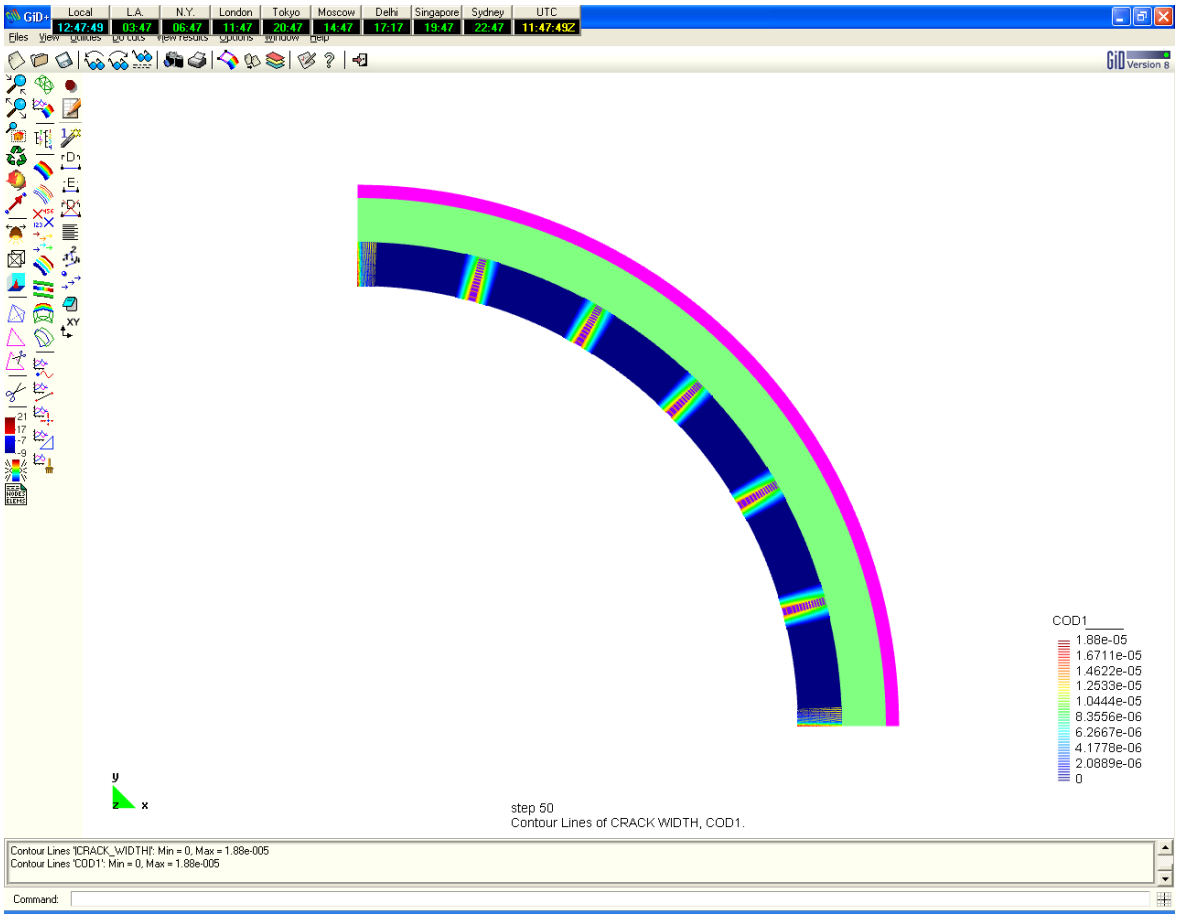

Figure 18: Displaying crack opening displacement isolines in GiD.

4.2 Postprocessing in ATENA 3D

The highest user comfort for post-processing is provided by Atena 3D. After executing ATENA 3D the result file for each increment can be loaded into the program by using the menu File | Open other | Results by step (*) … This command activates a dialog that can be used to load ATENA binary files with results to ATENA 3D postprocessor. When the user finishes loading the needed result file and closes the dialog, ATENA 3D postprocessor is automatically started. It should be noted that it is possible to open only binary result files that come from the same analysis. For further details about post-processing the user should consult the Part 2-2 of ATENA user's manual [2].

5. CONCLUSIONS

This tutorial provided a step by step introduction to performing combined thermal and stress analysis using ATENA software with GiD preprocessor. The tutorial involves an example of an axisymmetric pipe heated from the outside from 25 to 37°C in 10 hours. Then the heating remains constant for another 8 hours, and is afterwards cooled back to 25°C in 6 hours.

The objective of this tutorial is to provide the user with basic understanding of the program behavior and usage. For more information the user should consult the user's manual [2] or

contact the program distributor or developer. Our team is ready to answer your questions and help you to solve your problems.

The theoretical derivations and formulations that are used in the program are described in the theory manual [1].

Experienced users can also find useful information in the manual for the analysis module only [4].

6. LITERATURE

[1] ATENA Program Documentation, Part 1, ATENA Theory Manual, CERVENKA CONSULTING, 2009

[2] ATENA Program Documentation, Part 2-1 and 2-2, ATENA 2D and 3D User's Manual, CERVENKA CONSULTING, 2008

[3] ATENA Program Documentation, Part 3, ATENA 2D Examples of Application, CERVENKA CONSULTING, 2005

[4] ATENA Program Documentation, Part 6, ATENA Input File Format, CERVENKA CONSULTING, 2009

[5] ATENA Program Documentation, Part 7, AtenaWin Manual, CERVENKA CONSULTING, 2008

[6] ATENA Program Documentation, Part 8, User's Manual for ATENA-GiD Interface, CERVENKA CONSULTING, 2008